

513-39

198330

1-25
N94-22621

**NASA Langley Developments in Response Calculations
Needed for Failure and Life Prediction**

Jerrold M. Housner
NASA Langley Research Center
Hampton, VA

~~SECRET~~ INTENTIONAL

OUTLINE

- Computational methods for inserting detail in global models
 - Joining 2-D shell/plate structural regions
 - Joining 2-D shell/plate and 3-D structural regions
- Automatic adaptive refinement methods to identify areas requiring further detail analysis
- Rapid algorithms for large complex structural analysis

Page 286

PRECEDING PAGE BLANK NOT FILMED

287

C-4

STRUCTURAL FAILURE ANALYSIS IN CONCURRENT ENGINEERING

Successfully competing in the aerospace world market requires significant reductions in the time to design and develop aerospace systems. Competition also requires innovative concepts for using new material systems, achieving high-performance, attaining low development and maintenance costs and long-life designs. For example, although development of composite structural concepts began some twenty-five years ago, only now are composite materials seeing significant application in commercial aircraft and spacecraft structural systems. This lag is due in large part to the inability to predict with confidence the behavior and life of composite aerospace structures under a variety of environmental conditions. Often, details which are not accounted for in the preliminary stages of design become costly and time-consuming items if addressed later in the design process. Computational tools which will allow more unified and integrated designs in a concurrent engineering environment are also needed for more efficient treatment of important system behavior. These same computational tools can also be used to reduce costs associated with development, verification and certification tests. NASA's Computational Mechanics Branch at the Langley Research Center is helping to satisfy that need by developing validated, efficient, reliable and easy-to-use computational tools to be used in assessing failure and life of structural components.

- NASA's Computational Mechanics Branch is performing research to help meet some of the goals of concurrent engineering, namely,
 - Avoid costly redesigns late in the design process

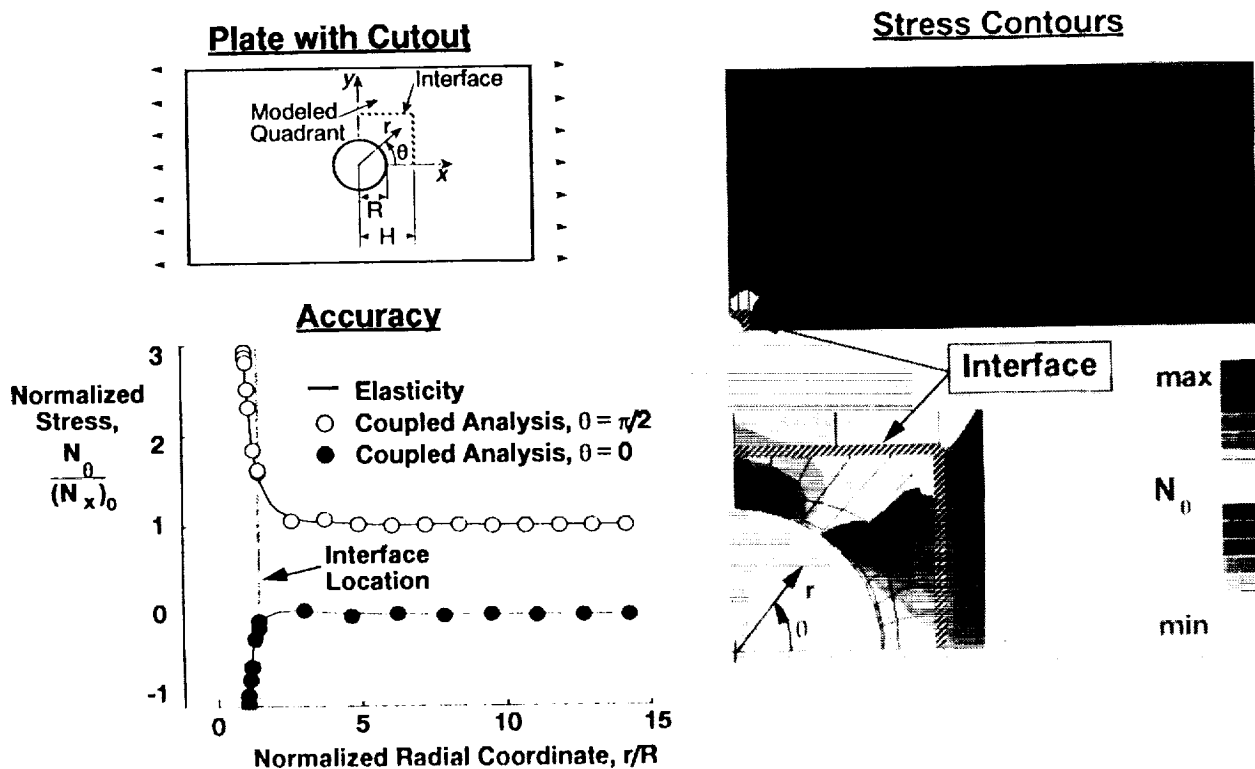
Details ignored in early design often lead to or become sites for pre-mature failure

 - Reduce costs associated with development, verification and certification tests
- Develop validated, reliable and easy-to-use computational failure analysis tools

ACCURACY OF INDEPENDENT REGIONAL MODELING DEMONSTRATED ON CLASSICAL EXAMPLE

New modeling technology has been developed to enhance detail global-local analysis. This modeling technology allows detail finite element models to be easily inserted within global finite element models. The salient feature of this technology, which simplifies detail insertion, is that the finite element grid points of the global and local regions need not coincide. This removes the often tedious construction of transition modeling to connect the global and local regions. The technology employs a specially developed interface element which has no width dimension. The interface element is based on hybrid variational principles of mechanics. The interface element has its own deformation shape functions and Lagrange multiplier functions which are used to impose compatibility between the regional models. From the engineer's viewpoint, the interface element is used just like any other finite element. Stiffness and mass matrices are automatically assembled including the effects of the interface connection.

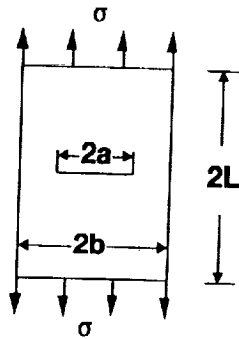
Before development of this technology was initiated, it was envisioned that the interface element would need to be placed away from steep stress gradient areas, that is, in areas where the gradients are fairly benign. However, the hybrid formulation does not require such a restriction on the placement of the interface. To demonstrate that, a classical case of an isotropic plate with a central circular cutout under uniform axial tension is examined. One quarter of the plate is analyzed. The interface between the global and local regions is placed very close to the edge of the cutout. In the accompanying figure it has been placed at 80% of the cutout radius which is at a steep stress gradient location. Nevertheless, the coupled global/local analysis using the interface element gives accurate stress resultant predictions as shown in the lower left quadrant of the figure for both N_x and N_y when compared to the elasticity solution. The lower right hand portion of the figure illustrates the color contours of axial stress resultant. It reveals stresses across the interface.



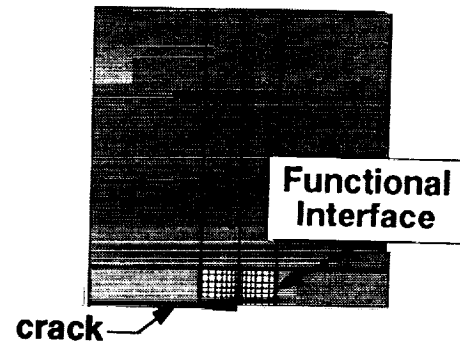
FUNCTIONAL INTERFACE METHOD ACCURATELY JOINS INCOMPATIBLE FINITE ELEMENT MODELS

The interface element can also be used to insert cracks into undamaged global models as shown. The accuracy of the method is shown by using the predicted stresses and displacement from the global/local coupled method to calculate stress intensity at the crack tip. Because no tedious transition meshing is needed to join the local and global regions, various crack lengths can easily be handled by moving the local region to the right as longer cracks are inserted or to the left as shorter cracks are inserted. Results are in excellent agreement with the reference solution for all crack lengths considered.

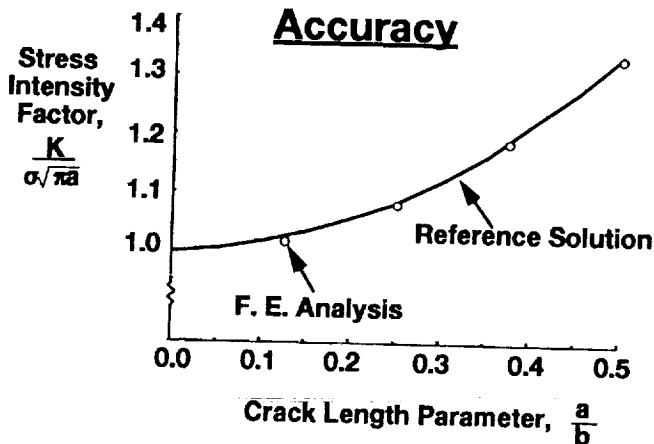
Cracked Plate



Quarter Model



Accuracy

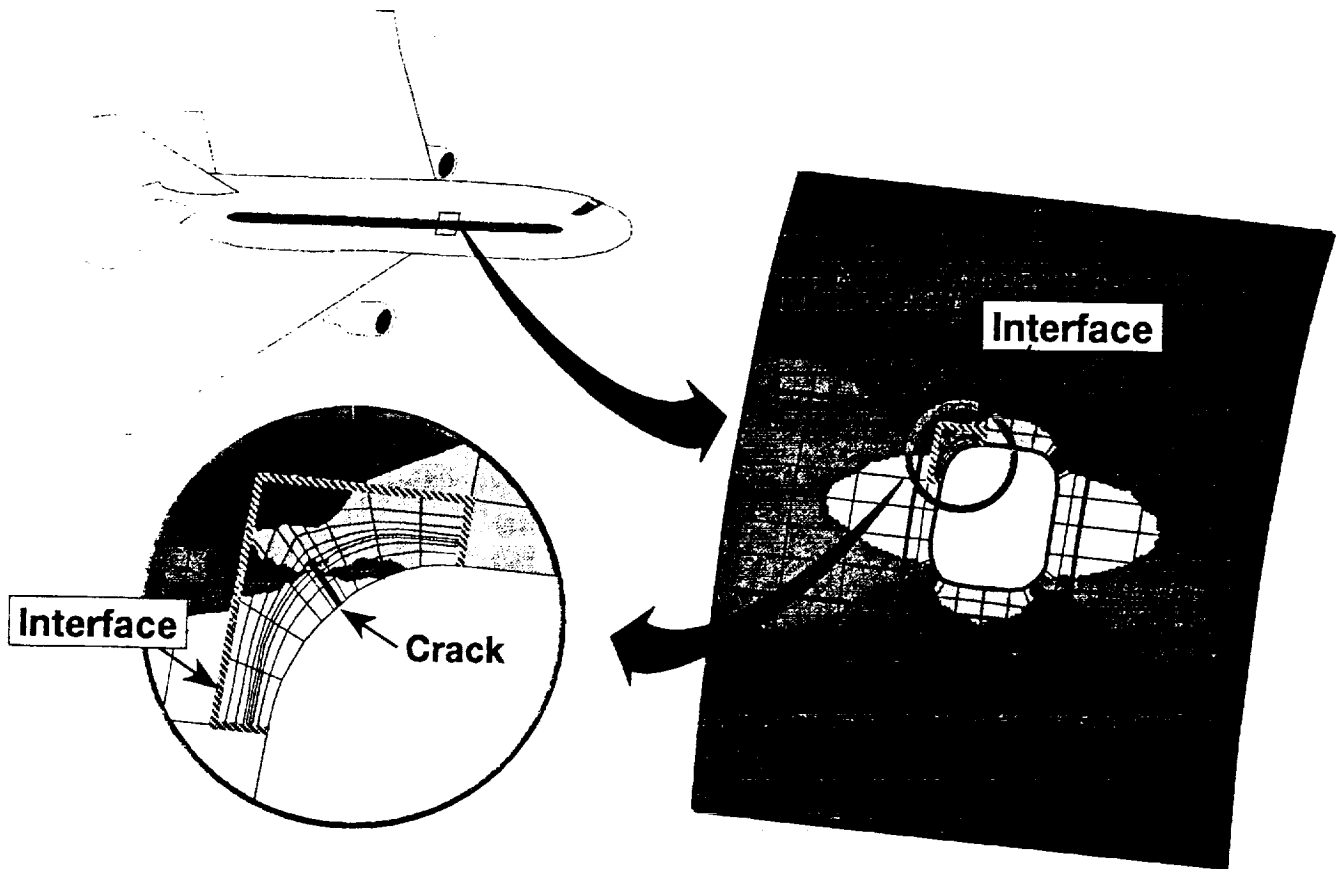


Advantages

- No Tedious Transition Region Modeling
- Grid Points Along Interface Need Not Coincide
- Requires Fewer Degrees of Freedom than Model with Transition Regions
- Retains Accuracy

INTERFACE METHOD FOR INSERTION OF LOCAL DETAIL MODELING EXTENDED TO CURVE PRESSURIZED FUSELAGE WINDOW PANEL

As an additional example on the utilization of interface elements, a global model of a panel in the window belt area of a fuselage is used. In order to assess the damage tolerance of the design, a crack is placed at the corner of the window. This is done by inserting a refined crack model into the global coarse model. Again, the color contours reveal the smoothness of the stresses across the interface.* Since no reference solution exists for this case, a refined finite element model was created and results from it compared with the results using the interface method. The stresses from the insertion modeling differ with the refined analysis by about 1%.



*Not shown in color.

INTERFACING CONCEPT FOR JOINING STRUCTURAL REGIONS

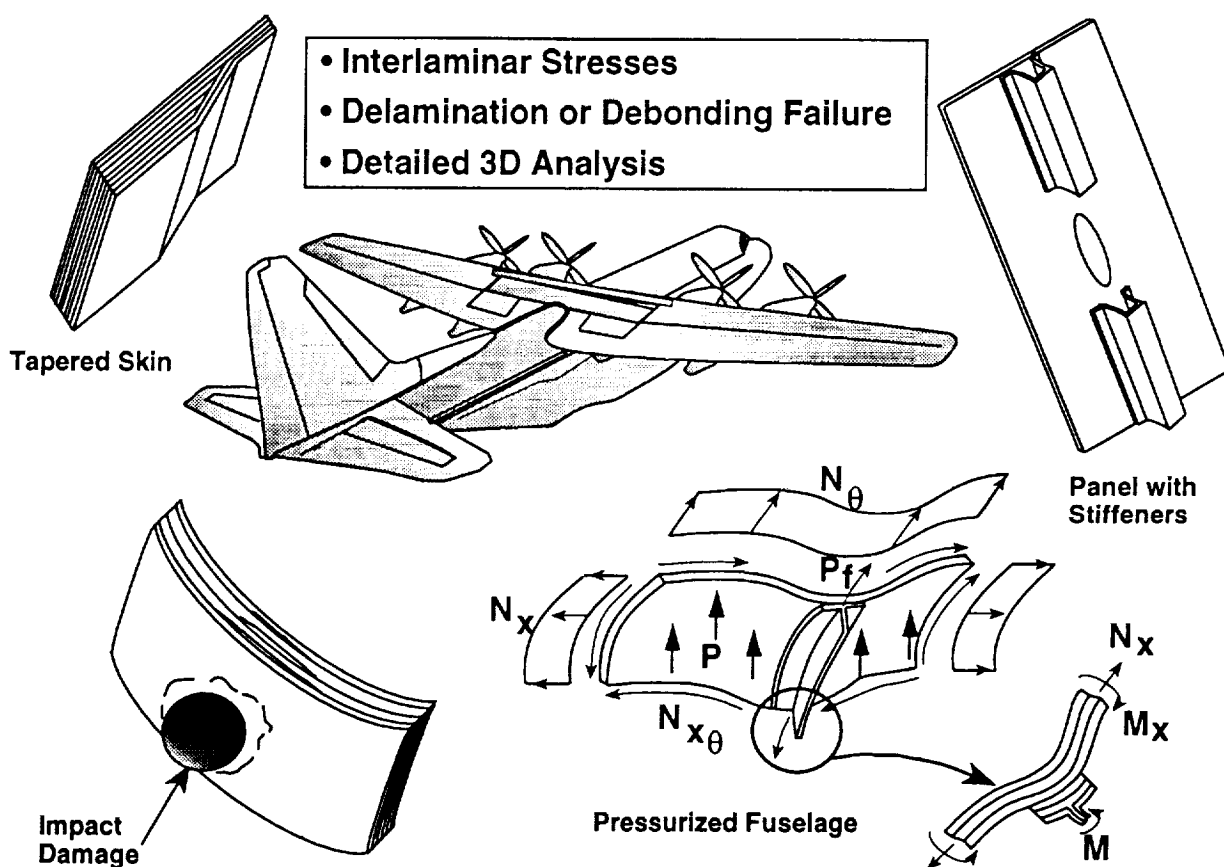
This chart summarizes the attributes of the interface element and regional joining technique. Basing the interface element on a hybrid variational principle is critical to the success of this technique. Other techniques, also based on variational principles, were developed and tested. These include a collocation technique (very similar to the commonly used multi-point constraints of general purpose finite element codes such as NASTRAN), and a least squares technique. In each technique a functional form, with free parameters, is laid down along the interface. In the collocation and least squares techniques this functional represents the displacements of the interface while in the hybrid technique two functionals are used, one for the displacements, and another for the Lagrange multipliers which enforce compatibility. Each region is tied to the interface using a collocation, least squares or hybrid integrated compatibility, respectively. The techniques were tried and their performance was compared on several test cases. It was found that the hybrid variational approach was consistently superior and gave accurate and reliable results. Because the regions never touch each other, except through the interface, grid coincidence is not required and models of regions or substructures can be created independently. This attribute of the method also avoids tedious-to-construct transition modeling between the refined gridded region and the coarse gridded region. This is extremely important for ease of detail refined model insertion in coarse global models and in substructuring, especially where structural components are developed independently by different individuals or organizations.

- Based on hybrid variational principles of mechanics
- Functional form for deformations and tractions is laid down along the interface
- Each structural region is joined to the interface through hybrid variational principles
 - Regional models can be developed independently of one another
 - Mesh grid points of adjoining regions need not be coincident
- Tedious-to-construct mesh transition regions between models are eliminated
- Retains accuracy while providing modeling simplification

MOTIVATION FOR COUPLED 2D-3D ANALYSIS

The use of composite materials for structural applications requires that detail which usually is not needed in monolithic metallic structures be considered. For example, in shell-type aerospace structures, through-the-thickness stresses are nominally small and therefore traditionally treated as a "secondary" effect in metallic structures. However, in composite materials they are no longer "secondary" in nature since through-the-thickness strengths are also small. Traditional practices may overlook these effects in the preliminary design leading to costly redesigns later or to compromises in performance. Consequently, three-dimensional analyses are often required to calculate interlaminar stresses for prediction of delamination or debonding failures at skin-stiffener interfaces. Typical critical regions are at joints, ply drop-offs in tapered skins, discontinuous stiffeners (or stiffener runout), regions of sharp stress gradient (such as near the frames in a pressurized fuselage), and in the area affected by impact damage.

Since detail 3D analyses usually require many degree-of-freedom, it is desirable to only use the 3D modeling where it is necessary and to use 2D plate and shell modeling elsewhere. This is the motivation for specialized elements that will permit 2D and 3D regions to be joined accurately.



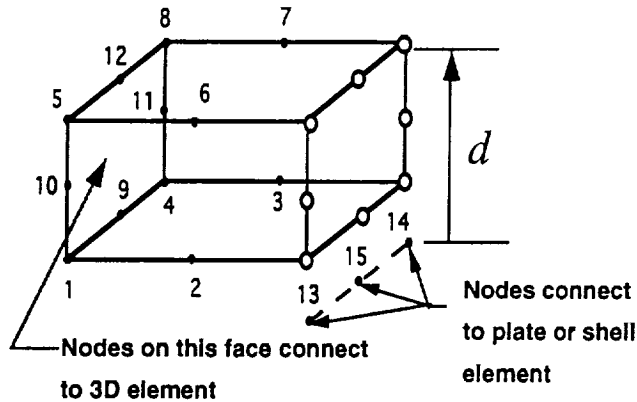
METHODS FOR COUPLING 2D MESH TO 3D MESH

Various methodologies have been proposed and experimented with to join 2D-3D regions. Among these are multipoint constraints and development of transition elements. The multi-point constraint approach requires considerable user input and is usually limited to linear constraints. The transition element approach is easy to use because it is treated like any other element and if done properly is applicable in nonlinear analysis. Recent developments in this area have led to transition elements which connect a stack of three-dimensional brick elements to a single shell or plate element. The methodology is also in place for elements which allow transition in two directions.

- Multipoint Constraint
 - Tedious modeling
 - Usually limited to linear constraints in general purpose FEM codes
- Transition Element (Element has solid nodes and shell nodes)
 - Surana (1980, 1982): Linear and nonlinear analyses
 - Liao, Reddy, and Engelstad (1988): Geometrically nonlinear analysis of laminated composites
 - Davila and Johnson (1991): Connecting a stack of solid elements to one shell element (linear and geometrically nonlinear analyses)

15-NODE TRANSITION ELEMENT

Illustrated here is a 15 node element which allows transition in one direction. Elements which allow transition in two directions are also available. Notice that the nodes connecting to the 2D plate or shell element may lie at a location off the element. This allows a stack of transition elements to be used so that a stack of three-dimensional bricks may be connected to a 2D plate or shell element. In a nonlinear analysis, the increments in the displacement variables represent the degrees-of-freedom.



Linear Analysis

$$\begin{bmatrix} u \\ v \\ w \end{bmatrix}_{Solid} = \begin{bmatrix} 1 & 0 & 0 & 0 & d \\ 0 & 1 & 0 & -d & 0 \\ 0 & 0 & 1 & 0 & 0 \end{bmatrix} \begin{bmatrix} u \\ v \\ w \\ \alpha \\ \beta \end{bmatrix}_{Shell}$$

Nonlinear Analysis

$$\begin{bmatrix} \Delta u \\ \Delta v \\ \Delta w \end{bmatrix}_{Solid} = \begin{bmatrix} 1 & 0 & 0 & df_1 & dg_1 \\ 0 & 1 & 0 & df_2 & dg_2 \\ 0 & 0 & 1 & df_3 & dg_3 \end{bmatrix} \begin{bmatrix} \Delta u \\ \Delta v \\ \Delta w \\ \Delta \alpha \\ \Delta \beta \end{bmatrix}_{Shell}$$

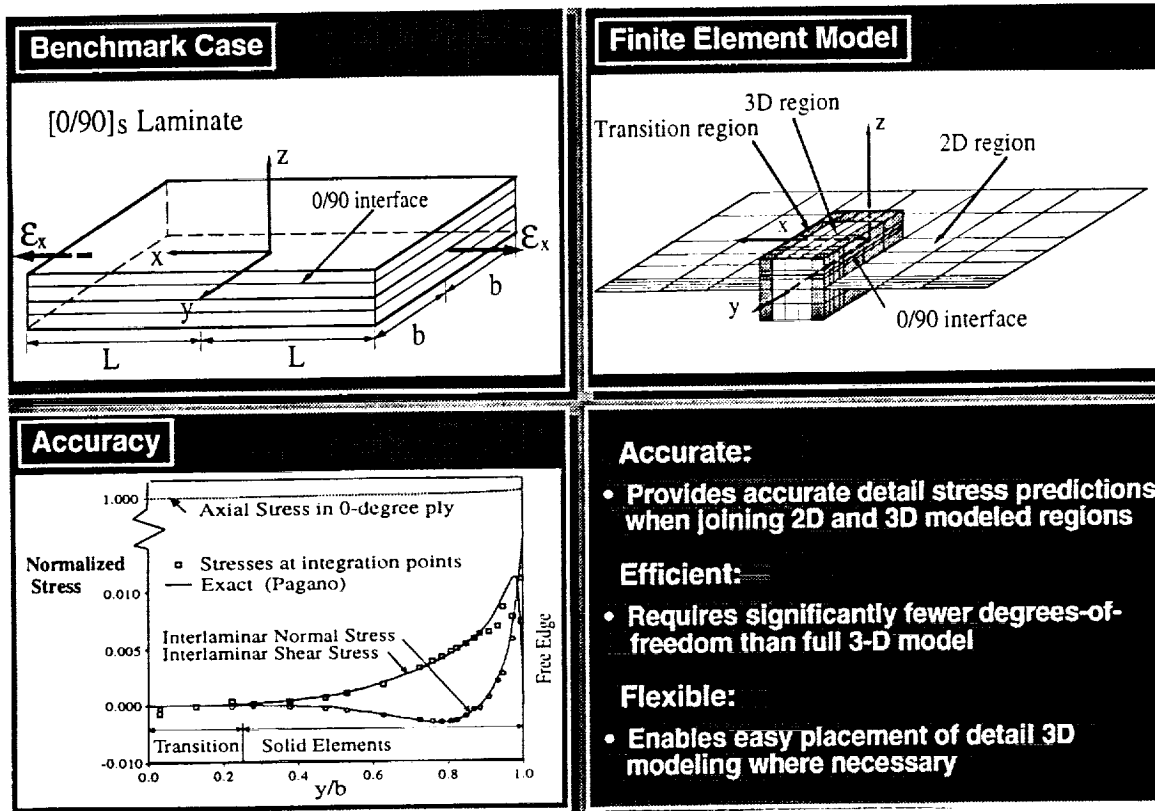
- f_m and g_n are trigonometric functions of accumulated rotations

- Ref.: Davila and Johnson (1991)

HIGH-ORDER TRANSITION ELEMENTS ACCURATELY CONNECT 2D AND 3D MODELED STRUCTURAL REGIONS

In this quad-chart a benchmark case of edge delamination analysis in a composite 0/90 symmetric laminate under tensile loading is used to display the accuracy of the 2D-3D transition elements. These transition elements use high-order anisoparametric shape functions for displacements and stresses. The displacement state as well as the transverse shear and normal through-the-thickness stresses are represented by high-order polynomials. These stresses are made compatible with the assumed displacement state in a least-squares sense. The high-order behavior of these transition elements prevents a pinching phenomenon which could otherwise occur due to the lack of through-the-thickness flexibility in the 2D elements. Such a pinching would overly constrain the 3D elements and lead to inaccurate results.

The 3D brick elements are placed in a region where the interlaminar stresses are desired. Even though the application case is symmetric, the bricks and associated transition elements have not been placed in a symmetric pattern about the x-axis so as to further examine the capability of the technique. As shown, predictions for the interlaminar stresses are in excellent agreement with the published solution. Also, because fewer degrees-of-freedom are required using a 2D-3D hybrid mesh rather than a full 3D mesh, the technique is efficient. The asymmetric placement of the 3D mesh indicates the flexibility of the technique to be able to place the detail 3D mesh where needed.



COMPRESSION PANEL WITH DISCONTINUOUS STIFFENER, COUPLED 2D-3D GLOBAL MODEL

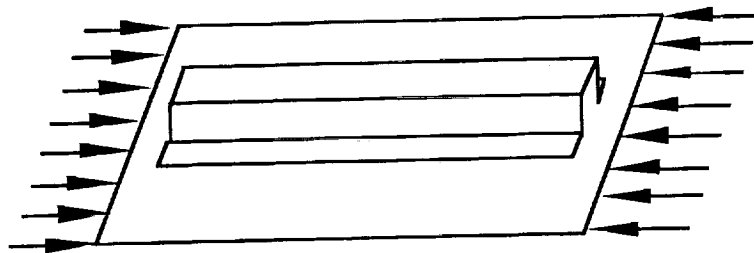
AND

AXIAL SURFACE STRAINS AT THE MIDDLE OF THE HAT STIFFENER

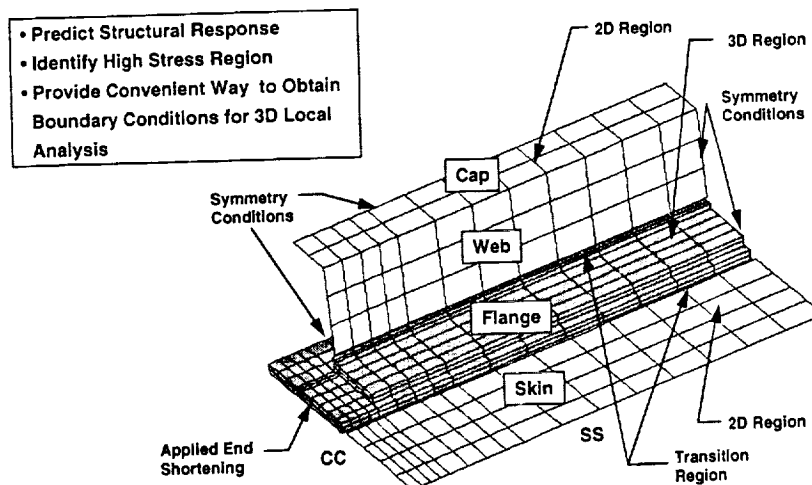
Discontinuous stiffeners are often found in practice as a result of cutouts and other reasons. The stiffener termination is often referred to as runout. Because of the discontinuity, severe stress states can exist at the termination of the stiffener. Illustrated here is a discontinuous hat stiffener. To examine possible skin-stiffener debonding, it is desirable to model the skin and stiffener flange with 3D brick elements and the rest of the skin as well as the web and cap of the hat stiffener with 2D plate elements. Taking advantage of symmetry, only one quarter of the panel is shown and analyzed.

The nonlinear growth in the predicted axial surface strains with increasing applied compressive load is shown at the middle of the hat stiffener. The use of nonlinear analysis for this example becomes apparent when the axial strains at the top of the hat are examined.

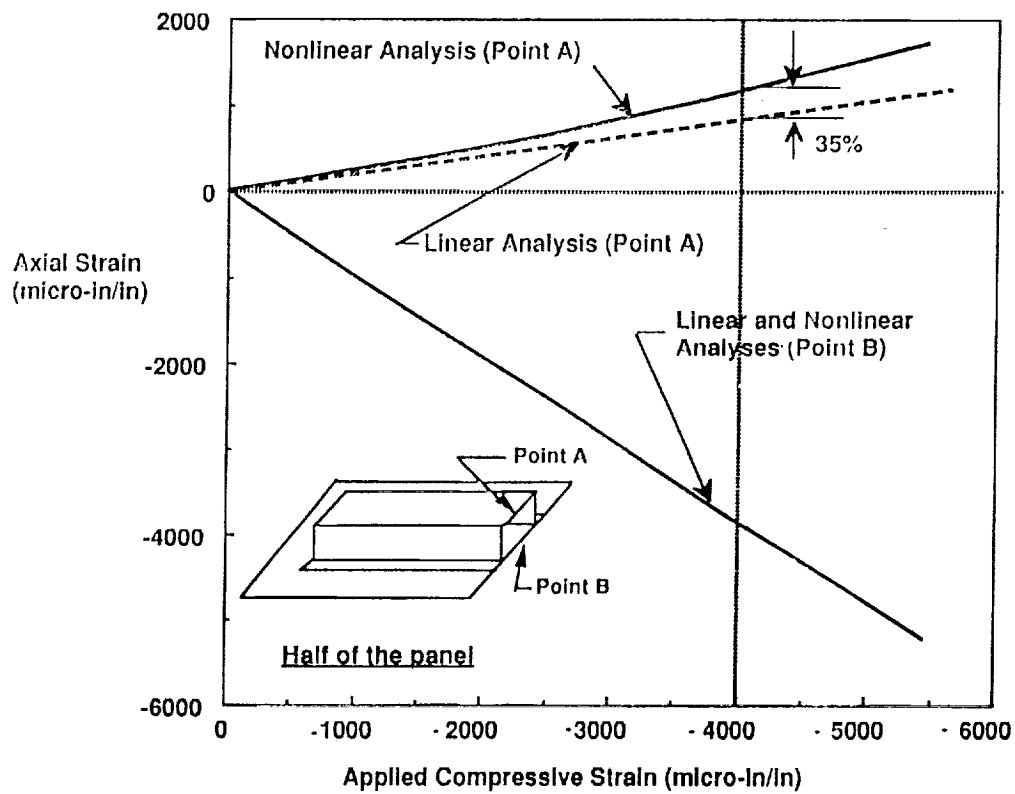
COMPRESSION PANEL WITH DISCONTINUOUS HAT STIFFENER



COUPLED 2D-3D GLOBAL MODEL



AXIAL SURFACE STRAINS AT THE MIDDLE OF THE HAT STIFFENER

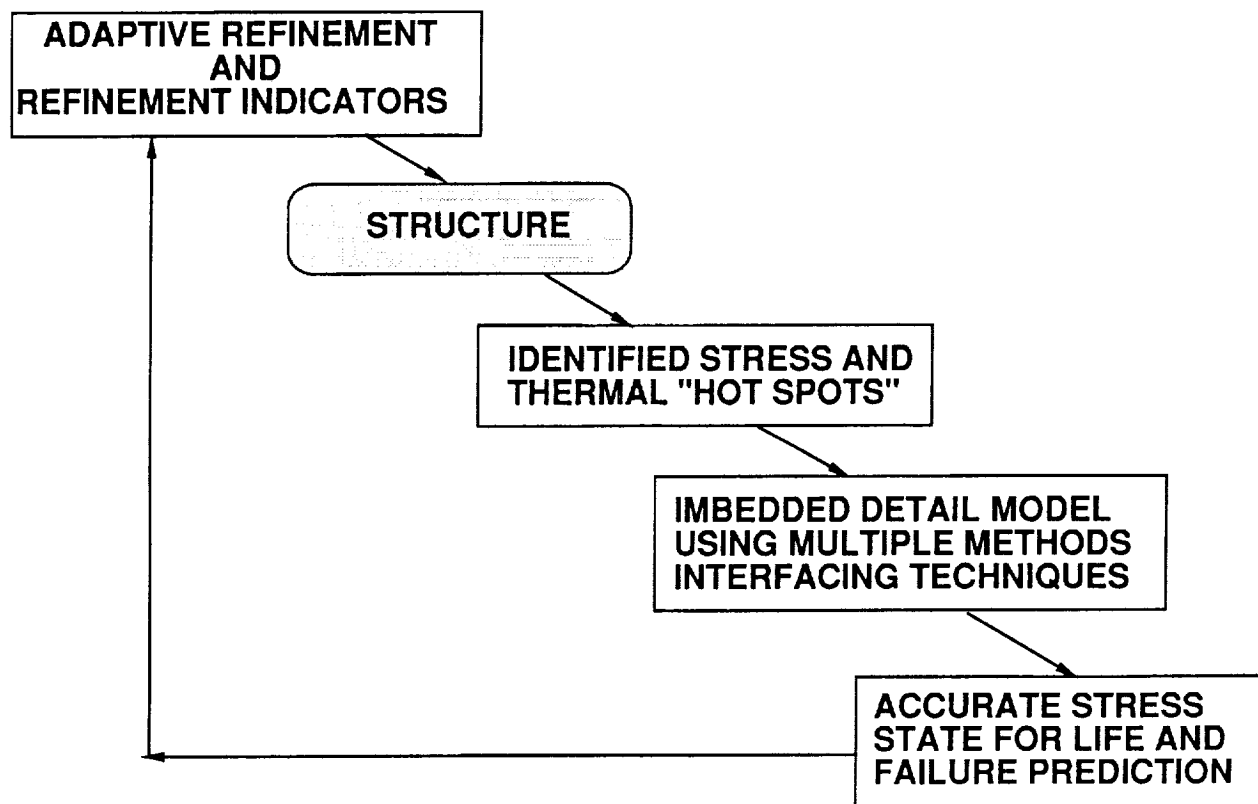


USE OF ADAPTIVE REFINEMENT WITH MULTIPLE METHODS

The ability to embed refined detail models in global coarse models by the use of interface elements and/or transition 2D-3D elements is a powerfully attractive capability for extracting the response behavior needed for failure and life predictive analysis. These techniques may also be referred to as multiple methods since they are a hybrid of two methodologies. Moreover, the interface element and transition element capability may be combined to connect structural regions and component aerospace substructures in three dimensions: the interface elements attending to the plate or shell surface and the transition elements attending to the plate or shell thickness. This capability becomes even more powerful when it is connected to adaptive refinement capability.

The accompanying chart illustrates how adaptive refinement which uses advanced refinement indicators can identify regions requiring further refinement. The imbedding techniques can then be judiciously applied at the identified locations.

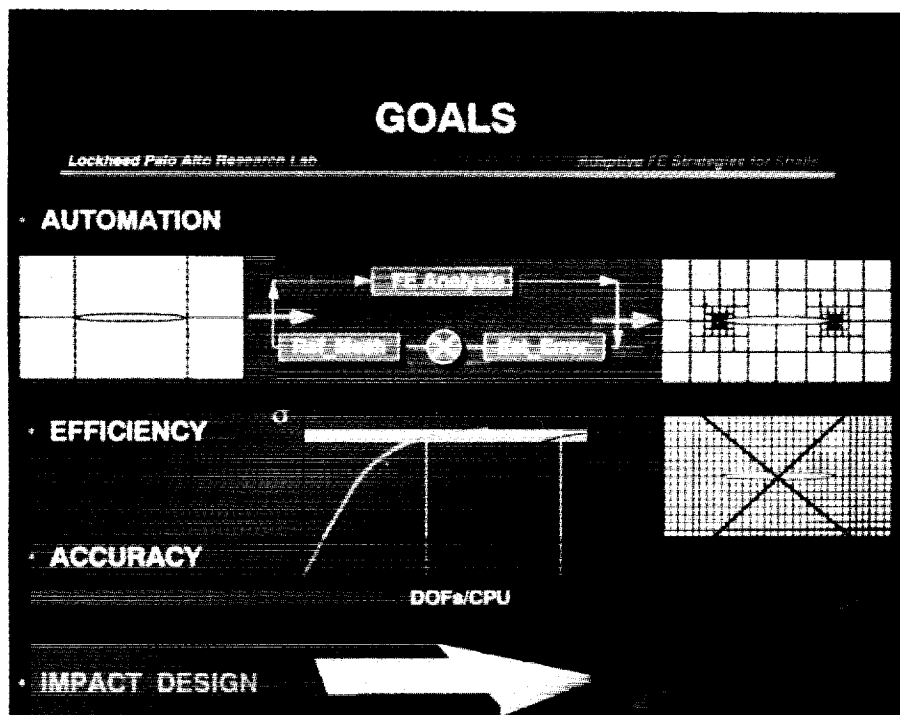
USE OF ADAPTIVE REFINEMENT WITH MULTIPLE METHODS



GOALS, CHALLENGES, AND APPROACH FOR ADAPTIVE MESH REFINEMENT TECHNOLOGY DEVELOPMENT

The goal of adaptive refinement is to provide a design tool for engineers which enhances finite element modeling by combining automation, efficiency and accuracy. Automation is brought about by the adaptivity of the mesh to highly stressed regions while accuracy and efficiency are brought about by using refined mesh only where necessary. Thus brute force global refined mesh over an entire component is avoided without the expenditure of much additional engineering time. The challenges to this technology are the treatment of (1) physical discontinuities where high stress gradients and usually stress singularities exist; (2) boundary layers; (3) components which are imperfection sensitive and hence mesh requirements can change dramatically with little change in imperfection; (4) adverse effects due to finite element distortion and element locking due to shell thinness which can require mesh refinements not driven by the physics of the response; and (5) geometric nonlinearity which requires mesh changes with increasing load and presents an enormous data handling challenge as well as procedures for mapping response from one mesh to another.

The COMET (COMputational MEchanics Testbed) code is used as a framework for developing adaptive mesh refinement capability. The modified code is denoted as COMET-AR. In this code an adaptive refinement procedure controls the code operation. The code consists of three major modules: an error estimator (or refinement indicator) module, a finite element analyzer, and a mesh refiner which operates upon certain built in rules and user-defined options. The mesh is refined until convergence is reached in accordance with a user-selected tolerance. An object-oriented data manager and database is an important feature of the code because of the intense data handling required in nonlinear adaptive refinement.

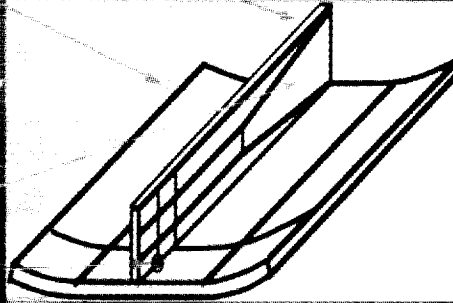


CHALLENGES

Lockheed Palo Alto Research Lab.

Adaptive FE Strategies for Shells

- PHYSICAL DISCONTINUITIES
- BOUNDARY LAYERS
- IMPERFECTION SENSITIVITY
- ELEMENT SENSITIVITIES
 - Distortion
 - Constraints
 - R/t
- GEOMETRIC NONLINEARITY

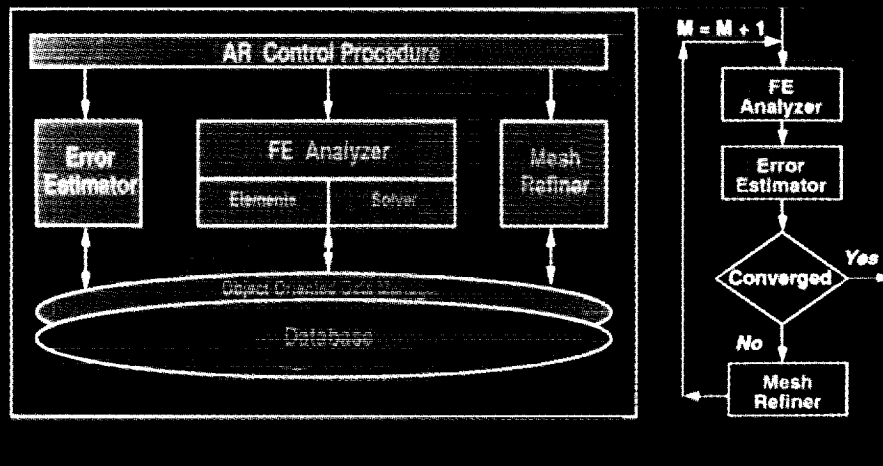


APPROACH

Lockheed Palo Alto Research Lab.

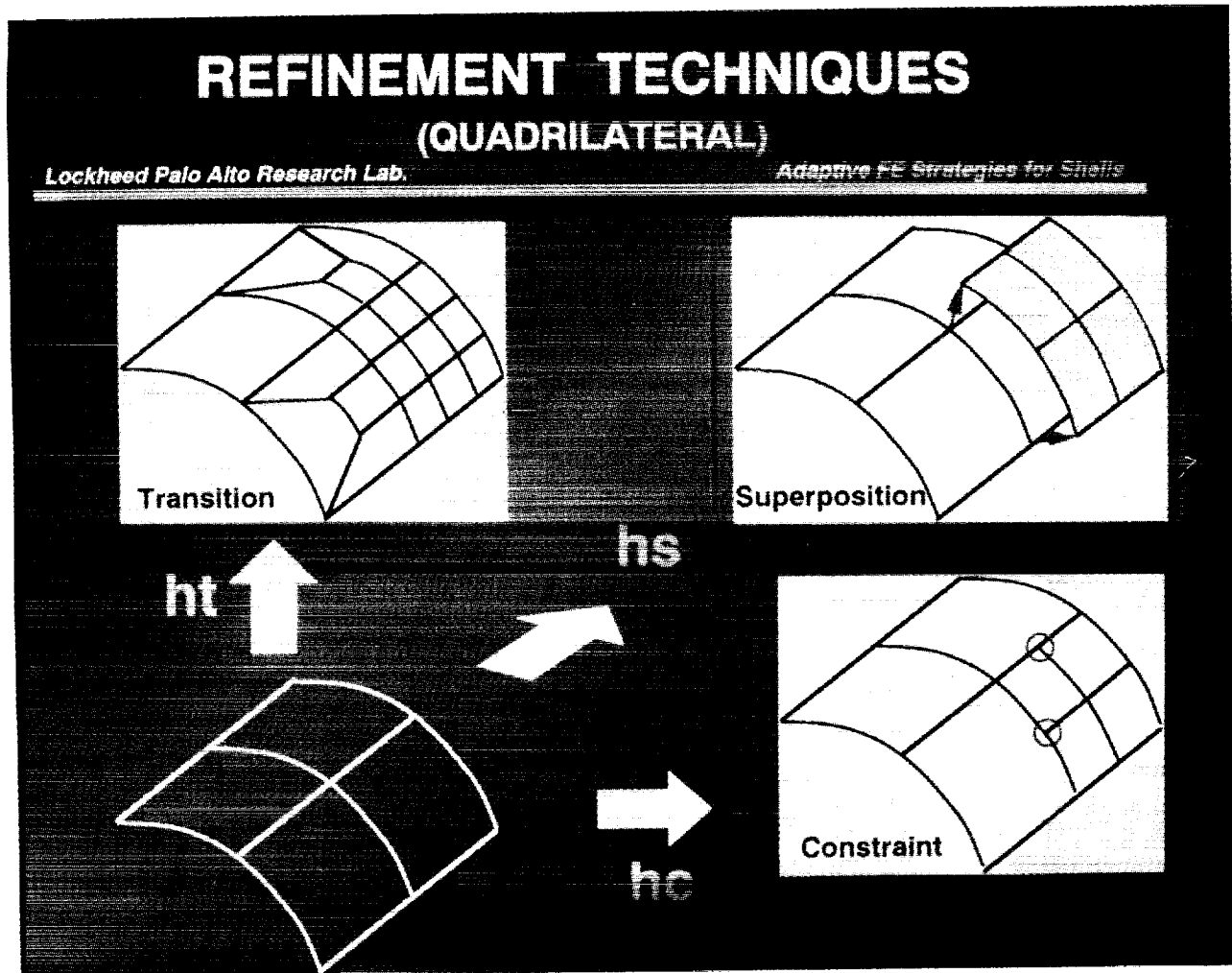
Adaptive FE Strategies for Shells

COMET (Computational Mechanics Testbed)



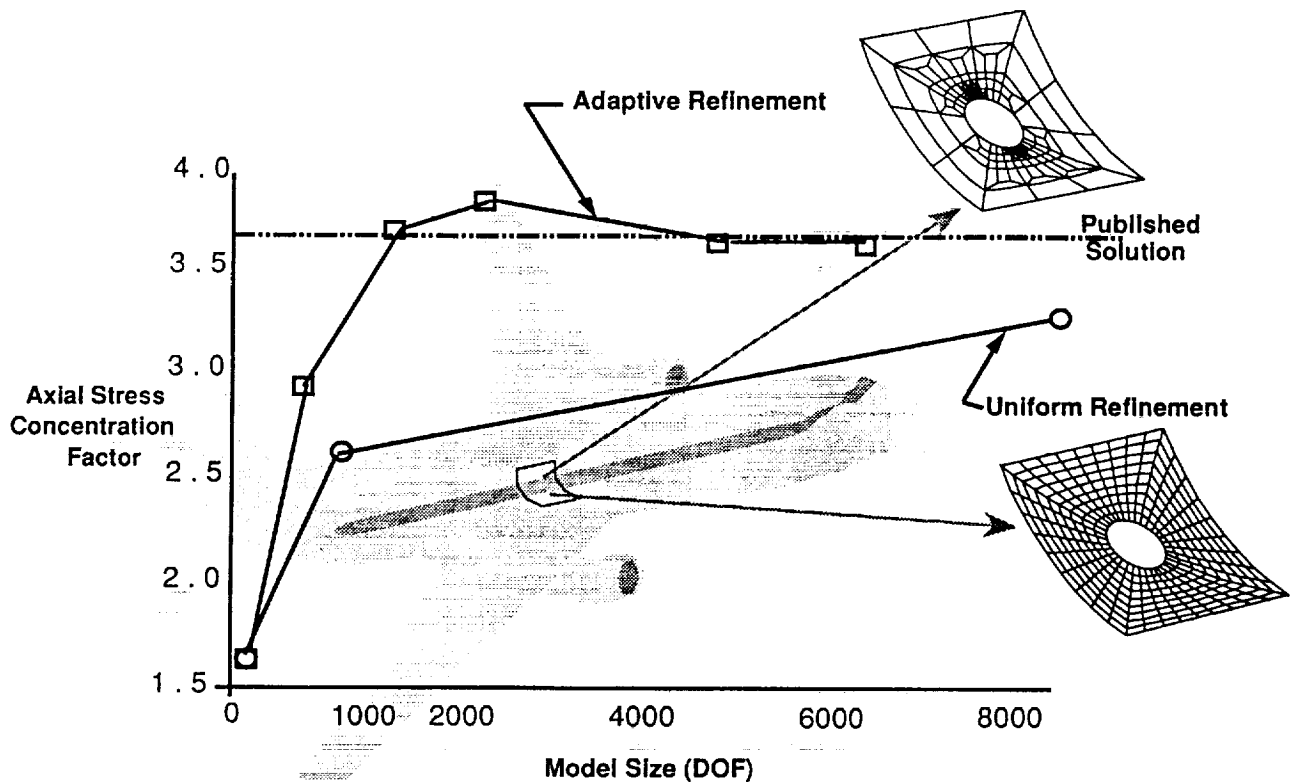
REFINEMENT TECHNIQUES FOR ADAPTIVE MESH REFINEMENT TECHNOLOGY DEVELOPMENT

Three different techniques are being explored, namely, transition refinement (ht), superposition refinement (hs) and constraint refinement (hc). Transition refinement tends to lead to distorted quadrilateral elements whose accuracy can be unsatisfactory. Superposition refinement builds upon regions of underlying mesh by refining in a hierarchical manner. It does not suffer from distorted finite elements provided the initial user-supplied coarse mesh does not contain distorted elements. The constraint refinement technique is mathematically equivalent to the superposition technique, but leads to a different form of the governing equations.



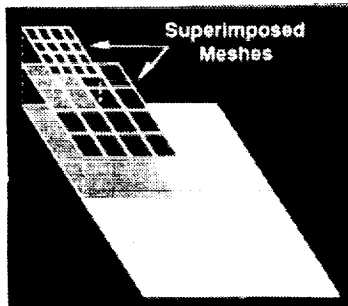
PERFORMANCE OF ADAPTIVE MESH REFINEMENT DEMONSTRATED ON COMPOSITE FUSELAGE-LIKE COMPRESSION PANEL

Convergence of automatic adaptive mesh refinement is displayed in the figure for a curved fuselage panel having a cutout. The panel loading is axial compression. The adaptive algorithm in the COMET-AR code zooms in by refining at the stress concentration generated by the presence of the cutout. Comparison of adaptivity results with those derived for uniform refinement, indicates the degree-of-freedom savings offered by adaptive refinement. However, the mesh refinement algorithm being used here does tend to generate distorted elements which, because of their poor accuracy can lead to over-refinement and in this case some over-prediction of the response until the mesh is further refined and convergence is reached.

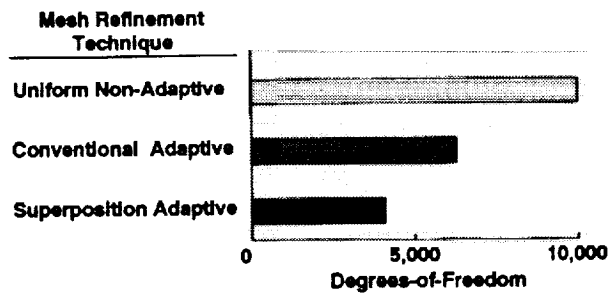


SUPERPOSITION ADAPTIVE REFINEMENT TECHNIQUE DEVELOPED FOR BUILT-UP STRUCTURES

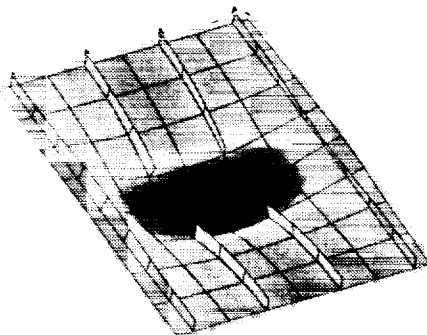
The advantage of the superposition technique (hs) over the conventional transitional technique (ht) is displayed in this quad-chart. The application case is a blade stiffened compression panel in which the blades are discontinuous. The ability of the adaptivity to capture the stress concentrations at the blade terminations is critical to the application of this technology to failure and life prediction. As shown by the color contours,* the axial stress concentrations at the stiffener runouts are well-captured with considerably fewer degrees-of-freedom than a uniform refinement and less than that of a conventional (ht) refinement. Notice that refinements are also taking place in the blade stiffeners as well as in the skin.



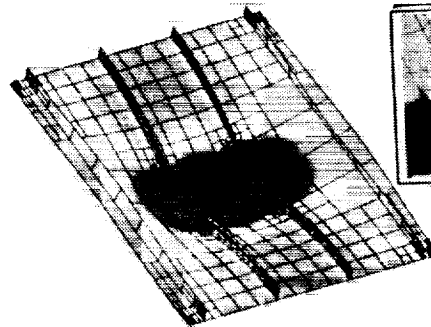
Superposition Method



Modeling Requirements



Initial Mesh



Final Mesh

Compression Panel with Discontinuous Blade Stiffeners

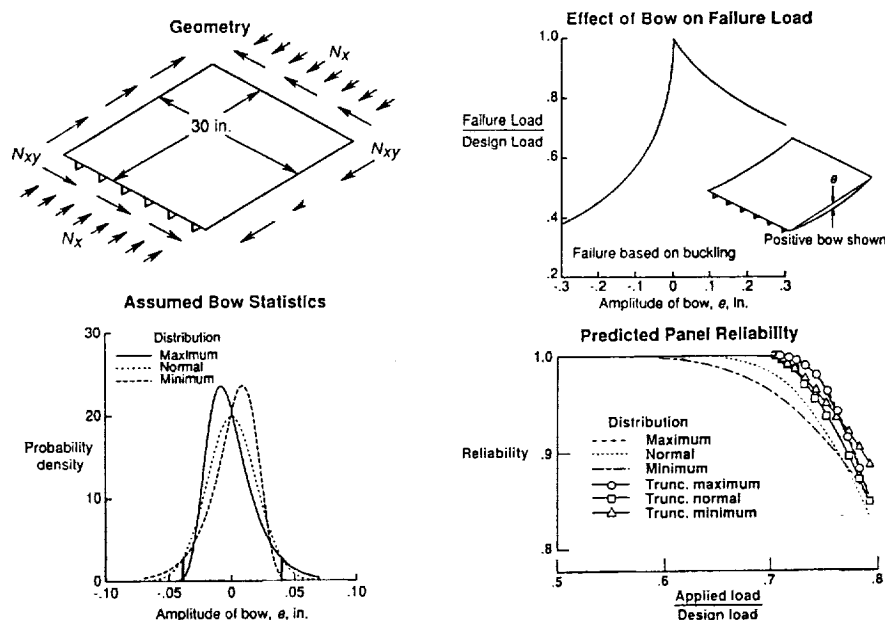


* Not shown in color.

STUDIES QUANTIFY EFFECT OF BOW-TYPE INITIAL IMPERFECTIONS ON RELIABILITY OF STIFFENED PANELS

A panel's reliability is the probability that it will not fail at under a specified load. That probability depends upon certain variables which vary from panel to panel. One such variable is the initial imperfection. Typically these imperfections are accounted for in design with safety factors. However, if these imperfections can be quantified statistically, then probabilistic methods can be used to calculate reliability. Understanding the factors affecting reliability can lead to improved design with lighter reliability. Studies were carried out to determine the reliability of stiffened panels assuming various reasonable probability distributions for the magnitude of a bow-type initial imperfection.

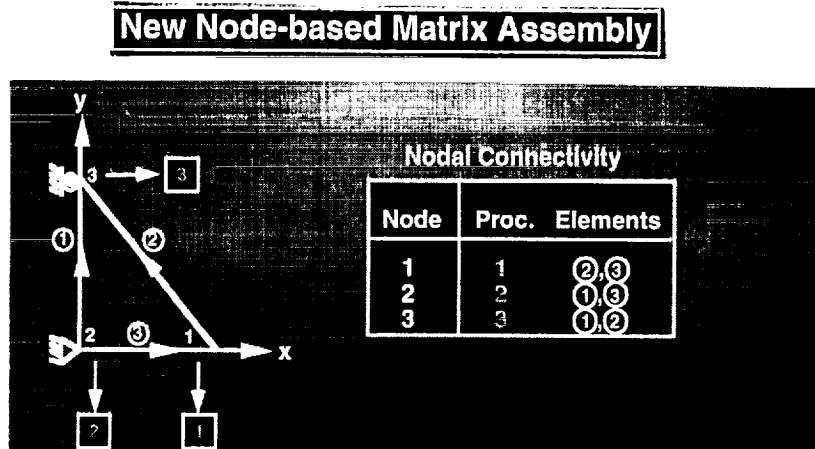
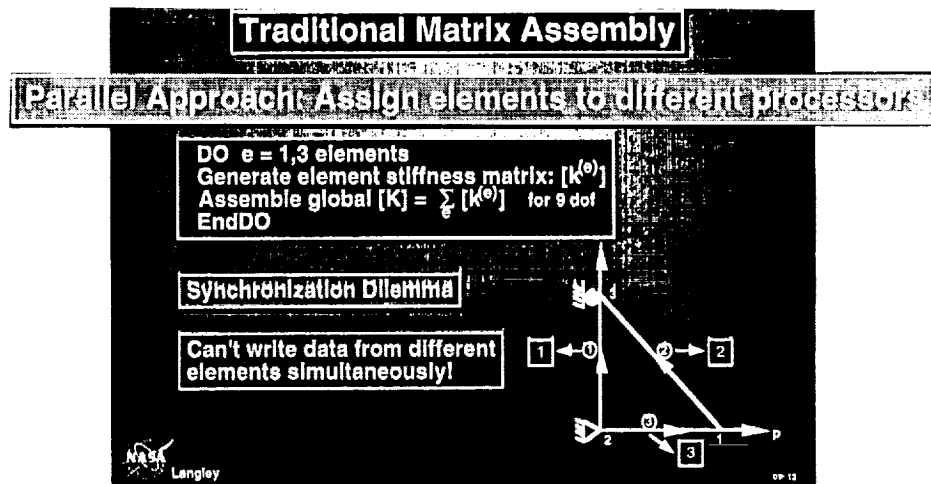
A graphite-epoxy, laminated, blade-stiffened panel having the configuration shown in the upper left-hand quadrant of the figure was designed to carry prescribed levels of combined compression and shear loads. The failure mode of the panel is buckling. The load carrying capability of each panel was calculated for various magnitudes of initial bow as shown in the upper right quadrant of the figure. The magnitude of the bow-type imperfection was then taken to be a random variable with various distributions of probability density functions as shown. Three distributions were considered: normal, maximum extreme value and minimum extreme value. Each of these distributions has a zero mean and the same standard deviation of 0.02 inches. The distributions only differ in higher moments about the mean. To simulate quality control, the distributions were truncated at plus or minus 0.04 inches. The reliability of the panel was calculated for all six distributions and all analyses were carried out using the panel sizing code, PASCO. Even though the three distributions without quality control had the same general shape, with the same mean and standard deviation, the panel reliabilities are quite different. This indicates the sensitivity of reliability to details of the probability distribution. In the case in which truncated distributions were used to simulate the effect of quality control, the reliability of the panels was not as sensitive to details of the distribution, with the reliability of all three dropping off from unity at about the same load. Moreover, for the distribution labeled Maximum, the reliability does not appear or benefit from quality control.



TRADITIONAL MATRIX ASSEMBLY AND NEW NODE-BASED MATRIX ASSEMBLY

Structural matrix generation and assembly can be a large fraction of finite element analysis time. This is especially true in design optimization and trade-off studies as well as in nonlinear analyses where generation and assembly must be performed many times. The traditional or conventional approach to generation and assembly of stiffness and mass matrices is to use an element-by-element based scheme wherein the code loops over all the elements. Such a conventional approach does not map well onto parallel computers. If the element calculations in such a scheme are assigned to separate processors, (as would be desirable in balancing the computational effort of the processors), a synchronization or communication bottleneck occurs as several processors, each of which are associated with a connected finite element, attempt to write stiffness or mass contributions to the same memory address (associated with a finite element node) at the same time.

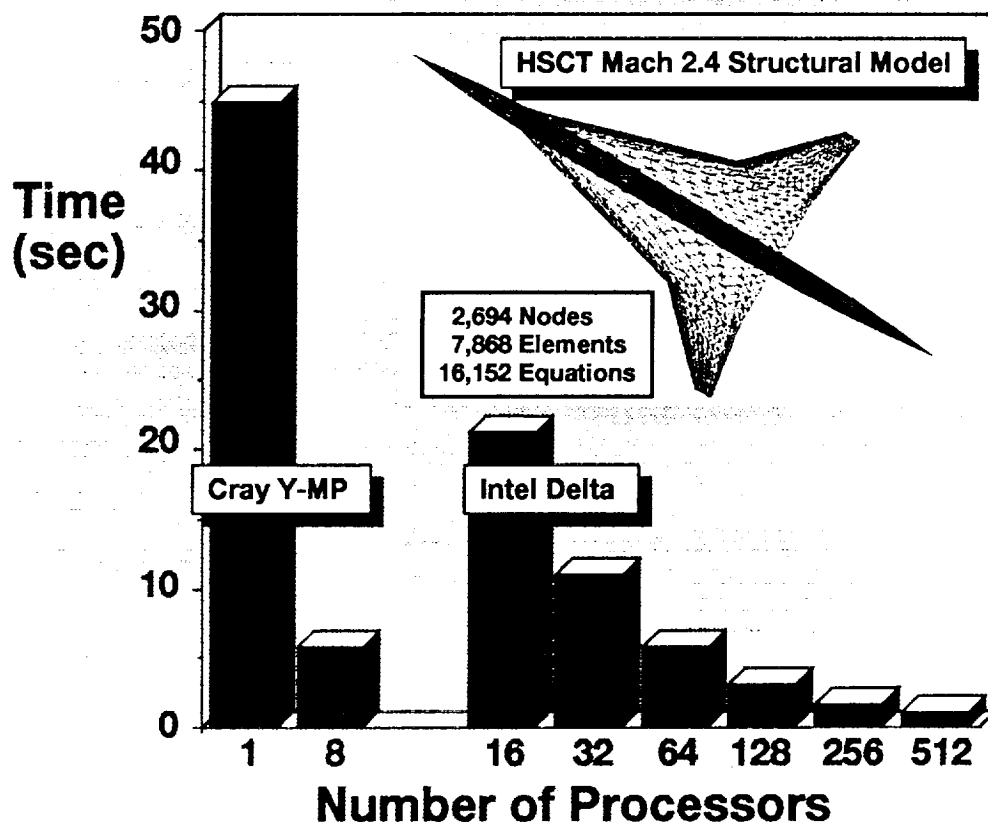
An alternative approach, which contains no synchronization bottleneck, is to loop over the nodes rather than the elements. This requires the creation of a map showing the elements connected to a node. Once that small overhead task is accomplished there is no communication required between processors and perfect speed-ups should be achievable.



STRUCTURAL MATRIX GENERATOR/ASSEMBLER FOR MASSIVELY PARALLEL COMPUTERS DEMONSTRATED ON HSCT

Conventional finite element codes, executing on sequential computers, use an element-by-element algorithm to generate and assemble stiffness and mass matrices. To parallelize this conventional procedure, element stiffness calculations are distributed among different processors. However, poor performance results since synchronization of these processors is required to simultaneously add stiffness contributions from different elements connected to the same node. To overcome this, a parallel node-by-node stiffness and mass matrix generation and assembly algorithm was developed to distribute nodal, rather than element, calculations to different processors. The algorithm's parallel performance was evaluated on a finite element model of a Mach 2.4 version of a High Speed Civil Transport. The model contains over 16,000 degrees-of-freedom. Results were run on a 512-processor Intel Delta parallel computer and compared with those generated using the conventional element-by-element algorithm on the supercomputer industry standard, namely, an 8-processor Cray Y-MP supercomputer.

The algorithm's performance was found to be scalable, which means the computation time reduces in direct proportion to the number of processors used. This is the highly desirable result sought, but in general is difficult to obtain on massively parallel computers due to communication time between processors. This achievement was possible by replacing the communication-intensive element-by-element algorithm with the node-by-node algorithm to eliminate interprocessor communication.

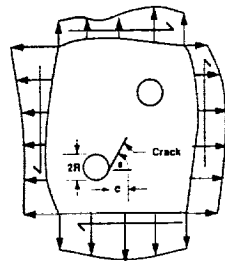


ITERATION METHOD DEVELOPED FOR EXTRACTING FRACTURE PARAMETERS

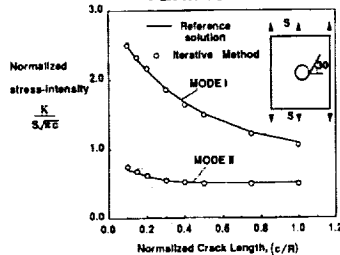
Rapid analysis tools for calculating fracture parameters are needed for predicting failure and life. Such a tool has been constructed for calculating fracture parameters (e.g., stress intensity factors) for cracks in general planar structural components. This tool uses a newly developed iterative method which combines the boundary element method (BEM) for an uncracked finite component with cutouts under general loading conditions with a continuum solution for a cracked infinite component without cutouts. A schematic illustrating the iterative method is shown in the accompanying chart. First the BEM is applied to the uncracked component. Since the BEM requires only the discretization of the component boundaries, (including cutout boundaries), the modeling is easier than with finite elements which require discretization of the entire component. Because the BEM step will not yield traction free conditions at the crack(s), the tractions predicted by the BEM step are removed by seeking the continuum solution for the equal and opposite tractions acting on the cracks of an infinite component. However, this results in unwanted tractions on the boundaries of the component. These are removed on the next iteration using the BEM. The iterative process continues until the BEM portion of the solution produces tractions on the crack surfaces which are negligibly small. The stress intensity factor for the crack is then the sum of the stress intensity factors obtained from all the iterations. Generally, only five to ten iterations are required to obtain converged solutions.

For verification, the iterative method has been applied to a plate under remote uniaxial tension having a crack emanating from a circular cutout in the plate for which accepted results appear in the literature. The variation of the stress intensity factor with crack length is shown for a crack emanating at 30 degrees from the transverse direction. Results are shown for Mode I and Mode II fracture; that is when the crack is driven by normal and shear stresses, respectively, at the crack tip. The excellent agreement with accepted results validates this new methodology which reduces modeling time and computer execution time.

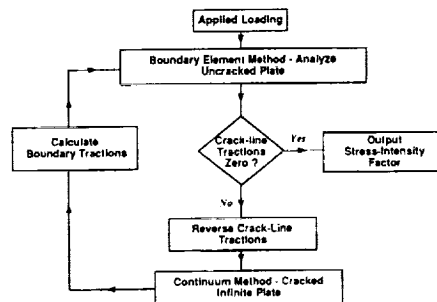
GENERAL 2-D CAPABILITY



VERIFICATION



ALGORITHM



EFFICIENCY

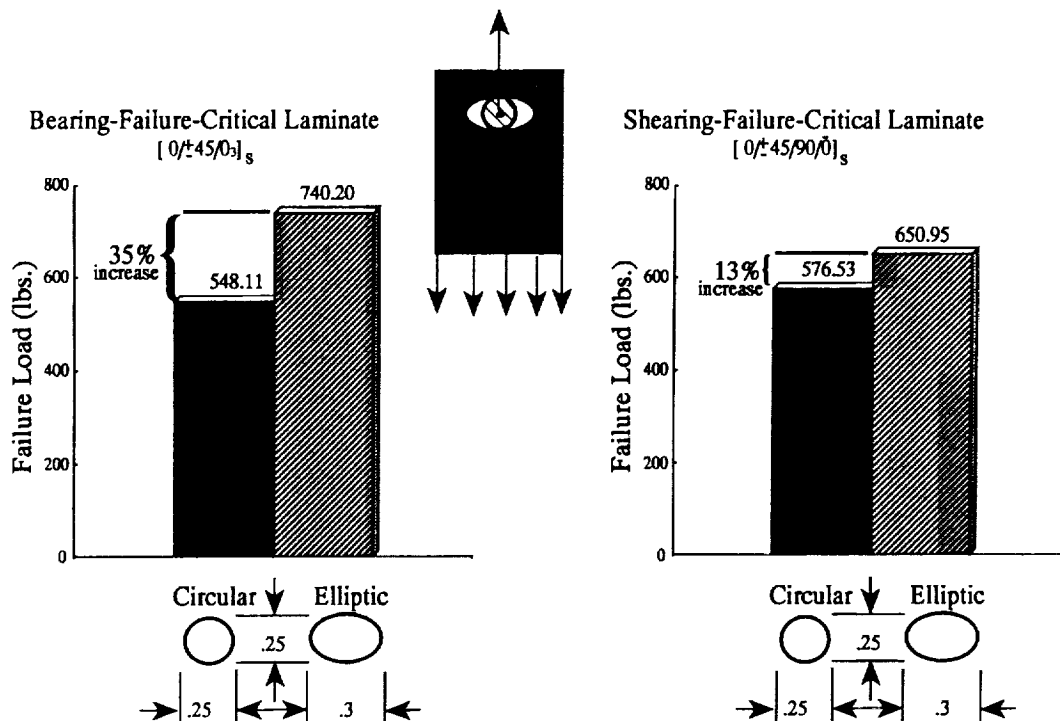
- Easy to Model
- Rapid Execution of Parametric Studies

POTENTIAL IMPROVEMENT IN ELLIPTIC BOLTED COMPOSITE JOINTS DEMONSTRATED

Composite structural components are often joined using circular bolts. These joints often fail in the composite laminate as a result of excessive bearing or shearing stresses. Increasing the bolt diameter reduces these stresses, but because it also reduces the distance from the edge of the bolt hole to the edge of the joint, it may not be practical. An alternative to reducing these stresses is to use an elliptic shaped bolt. Such a bolt may be fastened with an attached circular threaded shaft and the elliptic hole may be cut using water jet technology. To demonstrate the potential advantage of elliptic shaped bolts, analyses were performed. A closed-form approximate solution for bolt-loaded elliptical holes was formulated based on laminate theory and anisotropic elasticity. The normal load distribution on the edge of the elliptical hole was represented by a cosine series. Unknown coefficients of the cosine series were determined by a boundary collocation procedure in which the bolt is assumed to be rigid. A modified Tsai-Wu failure criterion was used to predict joint failure.

As demonstrated in the chart, two composite laminated constructions were studied. One was chosen because it is bearing-failure critical while the other is shearing-failure critical. Also, a circular bolt diameter of a quarter inch, typical of aircraft joints, was used. The elliptic bolt was chosen to retain the 0.25 inch dimension along the minor ellipse axis with a 0.30 inch dimension along the major axis. The chart shows a 35% strength improvement for the bearing-failure critical laminate design and a 13% strength improvement for the shearing-failure critical design. These results were predicted by the closed-form approximate solution and confirmed by a detailed finite element solution.

Elliptic shape bolts can be employed in design and re-design field modifications. If a joint design using circular bolts was found to be failing on some aircraft in service, the circular bolt holes on unfailed joints could be reshaped to accommodate elliptic bolts. This would constitute a relatively inexpensive fix.



SUMMARY

- Enhance failure analysis in concurrent engineering environment
 - Automatic adaptive refinement techniques to identify potential failure sites
 - Modeling methods to simplify treatment of failure precipitating locations
- Algorithms for high-performance massively parallel computers to provide rapid performance of
 - Repetitive analysis
 - Parametric studies
- Rapid computational tools for fracture and strength analysis and design